CUSTOMIZATION OF WING ANALYSIS

Wang Linjiang and Matthias Haupt

ABSTRACT

This article is concerned about the customization of creating a FEM model of a wing structure. The row and column method is used to number the structure parts(ribs, spars, skins), then the property parameters of all parts can be respectively inputted using the spreadsheet in which data can be inputted easily and correctly. A PCL(MSC/PATRAN Command Language under MSC/PATRAN 6.0) code is developed for the customization from constructing airfoil curves to creating the entire wing FEM model. A zigzag wing is demonstrated to verify the code. At last a VFW614 wing is analyzed from creating airfoil curves to the show of stresses that are calculated using MSC/NASTRAN 68. The results shows that this customization is very effective and efficient, it makes such a difficult work of creating a wing FEN model become much easier. This method can likewise be used to fuselage and other complicated structures.

I. INTRODUCTION

The structure of a wing is complicated. It consists of skins, ribs and spars of different geometry and materials. So it is a very hard work and costs a lot of time to manually create a finite element model of a wing. By using some preprocessors it can be easier, but is still tedious and easy to input wrong structure data. Now we try to find a new method to change this situation with the powerful MSC/PATRAN Command Language(PCL).

II. ANALYSIS METHOD

2.1 Wing structure character

Although complicated, a wing still has some regularities. It is composed of cordwise ribs and spanwise spars. We define the part of a rib between two neighbor spars, or the leading edge and the first spar, or the last spar and the trailing edge as a rib part, and the part of a spar between two neighbor ribs as a spar part. According to the engineering experiences we can consider that the property of a rib part is the same, this means, if the part is divided into finite elements, these elements have the same properties, that is the same material and thickness for plate element, and

---

Nanjing University of Aeronautics and Astronautics, PR China

the same cross-sectional area and material for rod element.

Usually different part has different property. In order to assign different properties to different parts of the ribs, we use column and row to number every rib part. Column means cordwise rib part number, row means spanwise rib number.

The same way is used to spar part, where column means cordwise spar number, row means spanwise spar part number.

Every skin part between two neighbor ribs and two neighbor spars is also numbered as above. Column means cordwise, and row means spanwise.

A part mentioned above is a part between two real ribs or two real spars, if a above part has significant variable properties, for example significant different plate thicknesses or cross-sectional area, then image spar or rib can be used to lessen the part and let a part have unvariable property.

2.2 Creating geometry

Before creating FEM model, the geometry of a wing must be at first determined. Using aircraft design program PrADO developed in the TU Braunschweig the coordinates of every point on wing skin can be calculated. According to the actual wing structure and finite element modeling, ribs and spars(real and image) are set. Then the airfoil points of all rib parts can be calculated by PrADO. Every rib part (including heading and trailing rib parts) has up and down two rib curves which are created by connecting the airfoil points. An array of three dimensions $A(m,n,l)$ is used to store the curves, which m is rib spanwise number(row), n is cordwise rib part number(column), $l=1$ means up curve, and $l=2$ is down curve.

Connecting end points of two spanwise neighbor rib curves, the up and down spar curves can be obtained. An array $B(m,n,l)$ is also used to store the curves.

And the pillar lines are obtained by connecting two end points of up and down rib curves of a rib part. where only a two dimension array $C(m,n)$ is needed.

The foregoing curves are used to create rod elements in order to model spar booms, rib booms, and pillars connecting rib web and spar web, and also to create surfaces of wing skin, rib web and spar web.

Fig.1 is a zigzag wing which is modified from a part of wing VFW614, it shows the shrink curves of ribs, spars, and pillars, and the shrink surfaces of skin, rib, and spar.

Fig.1 The shrink geometry of a zigzag wing

It is important for a rib that rib parts must be fully set from heading edge through last spar or trailing edge. If there is no real part somewhere, image part must be set, otherwise spar curves would be wrongly connected, for example without heading rib part of a rib, the first spar curve of a neighbor rib will be wrongly connected to the second spar position of that rib. Because rib part is numbered sequentially according to inputted data, and spar curves are connected between two end points of two spanwise neighbor rib parts with the same chordwise number.

Skin surfaces, rib web surfaces, and spar web surfaces are created on the basis of curves. They are used to create plate elements and are respectively stored in arrays $D(m,n,l)$, $E(m,n)$, and $F(m,n)$.

2.3 Creating FEM model

On the basis of geometry model, a FEM model can be easily created. At first mesh seeds are created on all rib, spar, and pillar curves. According to the mesh seeds on the curve and the inputted material and cross-sectional area properties, corresponding rod elements are created. For image curves no properties are inputted, also no elements are created. Plate elements of the skin, rib, and spar are created in the same way. If no properties are inputted, no elements are created, this means openings. All nodes corresponding to the elements are created simultaneously.

Pressure loads are assumed to be applied on the up skin elements, and the elements on a skin part have the same pressure. Concentrated Loads F_x, F_y, F_z, M_x, M_y, M_z are applied on the up cross point of rib and spar. Boundary conditions can be applied on the nodes of up and down rib rod elements. Fig.2 is the shrink FEM model of Fig.1, there are openings on the up skin, rib and spar webs.

Fig.2 Zigzag wing FEM model

After all equivalencing is performed to reduce all nodes that coexist at a point to a single node, and optimization is done to minimum CPU time, memory, and disk space of the FEM problem.

III. CUSTOMIZATION

The above analysis is about the method of the creation of a wing FEM model. Using PCL(MSC/PATRAN Command Language), we developed a customization under MSC/PATRAN 6.0(Analysis preference is MSC/NASTRAN) to perform this process as following:

3.2 Start the customization

After starting the MSC/PATRAN, the main window of MSC/PATRAN(Fig.3) appears.The item “Wing” on the top bar menu is our customization button.

Fig.3 MSC/PATRAN main window

3.2 Start wing analysis

After clicking "Wing", a pulldown menu with only one item "Wing Analysis" will appear. Then click this item, the main form of wing analysis appears(Fig.4). There are three frames in the form. The first frame is to spawn a remote process to generate the points of the rib curves of the wing. The second frame is to create geometry of the wing. The third frame is to create FEM model.

Fig.4 The main form of wing analysis

3.3 Spawn FORTRAN process

Click the button "Start", start to spawn a remote process to generate the points on all rib curves of the wing.

3.5 Creating wing geometry

There is a default file in the databox. if another file name is expected, by clicking "Browse..." a file selection form appears, then select the expected file name. The file has a special format, it can be produced by the above spawning process, or by other methods yourself. After selecting the file, click the button "Start", MSC/PATRAN creates automatically all curves of ribs, spars, and pillars, and all surfaces of skins, rib webs, and spar webs. Very easy!

To verify the code the zigzag wing in Fig.1 is performed. It runs at workstation HP9000/C180(following is the same). 488 points, 116 curves and 71 surfaces are produced, it takes 19 seconds.

3.6 Creating FEM model

Inputing the FEM data is the most difficult and tedious work, and easy to make mistakes, but now we can easily do this. From Fig.7 we can see that there are nine buttons in this frame. The functions of up six buttons are similar.

At first click the button "Mesh Seed...", then the corresponding form (Fig.5) appears, a very big form which contains switches, frames, databox, optionmenus, labelicon, spreadsheet, and buttons.

Fig.5 Create mesh seed form

At the top there are three switch items, only and always one is ON, that means the current input status(Rib Curve, Spar Line, or Spar Pillar).

The Curvature Based Mesh Seed is used to set mesh seed on the curves. "Element Edge Length", "Target Element Order", "Allowable Curvature Error", and "Element Constraints" are all mesh seed parameters that can be chosen to create different mesh seeds(detail see User's Manual, Part 2:Finite Element Modeling, page 2--23). The data in databoxes can be changed by entering new values.

The important part of this form is the spreadsheet which contains columns, rows, column labels, row labels, and cells. As mentioned in §2.1, row means spanwise rib number or spar part number, column is chordwise spar number or rib part number. Cells are the corresponding curves of ribs, spars and pillars. For different input status there are different rows, for example rib curve status have one more row than spar curve and pillar status, they can be automatically adjusted.

After the mesh seed parameters are set, they can be set to the corresponding cells, there are following flexible methods to do this: 1) Click the button "Apply" to select all cells which means all cells have the same parameters. 2) Click row label to select entire row which mean the all cells of the row have the same parameters. 3) Likewise click column label to select entire column. 4) Click and not lose the mouse in the starting cell, and move the mouse to the ending cell, then lose the mouse to select a rectangular area, for one single cell only click this cell. All of the above four methods don't change the cells with "EEEEEEEEEEEE" that mean no geometry element here(see Fig. 3).

If some cells need to be corrected, the buttons "Clear selected" and "Clear All" can be used, then change the parameters and input them to the cells again.

After all of the cells are filled, click the button "Save" to save them into the memory, then click next switch item on the top, and go to the next curve status. If not all of the cells are filled, a caution form will appear to tell which cell is still blank in the information box, this is only for mesh seed form, for other forms the cells can be blank, this means that there are no elements or loads.

After all of the curves are done and saved, then click the button "Close" to close the form. So what a easy interface is it!

There are similar forms f or the "Plate Element..."(here is QUAD4),"Rod Element..."(here is ROD), "Conce. Loads...", "Pressure...", and "Boundary...". For elements the materials are connected with the material library in the database. There are up to five load cases for load and boundary condition forms. Concentrated loads are applied on the cross point of the rib and the spar of the up skin. Pressure loads are applied on the up skin. Boundary conditions are applied on the all nodes of the entire rib curve. Certainly you can only fill the cells which have loads or boundary conditions.

After all the data are inputted, they can be saved as a file by clicking the button "Save as...". The button "Read Saved File..." is to input saved file for the same or similar structures with different properties, then only some data need to be modified, and a lot of time can be saved. It is especially useful for design sensitivity study and optimization of a complicated structure.

Finally click the button "Start" to automatically and quickly create the FEM model. (mesh seed -- plate element -- rod element -- pressure loads -- concentrated loads -- boundary condition --load case -- node equivalencing -- node optimization).

The FEM model in Fig.2 is produced, it has 230 rod elements, 354 plate elements and one loadcase, after equivalencing there are 312 nodes. It runs about 4.5 minutes.

In order to observe the model clearly many groups(points, curves, surfaces, rod elements, and plate elements) are created during the procedure.

3.7 Creating data file for MSC/NASTRAN

At last click the Analysis button on the MSC/PATRAN main form to create data file for

MSC/NASTRAN. After the calculation in MSC/NASTRAN the results can be shown on the model.

IV. EXAMPLE

We take VFW614 wing as an example. There are 20 ribs and three spars. The half span of the wing is 10.590 meters. Rib 1 is the symmetric line of the whole wing, and real wing structure is from rib 2 to rib 20. The geometry of the wing has 1364 points, 341 curves, and 236 surfaces. It runs 55 seconds to create the geometry. The points are produced from PrADO.

The corresponding FEM model without up skin elements is in Fig.6. As an example it is simply assumed that skin thickness varies from 6 mm to 2 mm spanwise, from rib 2 to 6 is 6mm, from 6 to 13 is 4mm, and from 13 to 20 is 2mm. All rib webs and spar webs are 2 mm thick. All cross sectional areas of rod elements are 600mm^2. There is only one load case which has concentrated loads and pressure loads on the wing tip. Three tip points are applied with F_z=300N, and one tip surface part is loaded with pressure P=2100MPa and the root of the wing is constrained. The material is aluminium E=72.0GPa, μ =0.33. The model has 775 plate elements, 506 rod elements, 585 nodes, one load case. It runs about 15 minutes to create the model. Some image rib parts are set between spar 2 and 3, no real rib parts are there.

Fig.6 VFW614 wing FEM model without up skin element

The static analysis of this model is performed using MSC/NASTRAN V68 which lasts 38.4 seconds CPU and 4 minutes 22 seconds link time at workstation HP9000/730. Fig.7 shows the Von Mises stress distribution of the up skin plate element in deformed structure, undeformed structure is also printed as reference.

Fig.7 Von Mises stresses of up skin

V. CONCLUSION

The above methods and results show that the FEM model of a complicated wing structure can be created easily, quickly and correctly using this Wing Customization. Likewise with the powerful PCL other complicated structures, for example a fuselage, can also be customized. It is especially useful for design sensitivity and optimization of a complicated structure.

VI. REFERENCES

1. MSC/MSC/PATRAN User's Manual, PART 1: Introduction to MSC/PATRAN
2. MSC/MSC/PATRAN User's Manual, PART 2: Basic Functions
3. MSC/MSC/PATRAN User's Manual, PART 3: Geometry Modeling
4. MSC/MSC/PATRAN User's Manual, PART 4: Finite Element Modeling

5. MSC/MSC/PATRAN User's Manual, PART 5: Functional Assignments
6. MSC/MSC/PATRAN User's Manual, PART 6: Analysis Application
7. MSC/MSC/PATRAN User's Manual, PART 7: Results Postprocessing
8. MSC/MSC/PATRAN User's Manual, PART 9: PCL and Customization
9. MSC/MSC/PATRAN User's Manual, PART 10: Example Problems
10. PCL Reference Manual
11. MSC/MSC/PATRAN Preference Guides, MSC/MSC/NASTRAN